Physical Education, Health and Social Sciences
https://journal-of-social-education.org
E-ISSN: 2958-5996
P-ISSN: 2958-5988

Computational Investigation of Incompressible Laminar Flow in T-type Channel Fixed with Circular Obstacle at The Center with No-slip Boundary Condition

Akhtiar Ali Mirbahar[1,] Saeedullah Chachar[2], Muhammad Tarique Indhar[3], Rafique Ahmed Soomro[4], Asif Ali Lashari[5], Dost Muhammad Mako[6], Muhammad Bachal Korai[7,] Bakhtiar Ali Mirbahar[8]

[1]Assistant Professor Mathematics, Government Degree College, Pano Akil
Email: akhtiararab1981@gmail.com
[2]MS/Phil. Mathematics, Sukkur IBA University, Sukkur, Sindh, Pakistan
Email: saeedullah.msmaths21@iba-suk.edu.pk
[3]Ph.D. Scholar, Mathematics, Shah Abdul Latif University, Khairpur Mirs
Email: mtariqueindhar@gmail.com
[4]Associate Professor Government Islamia Science College Sukkur
Email: rafiquesoomro05@gmail.com
[5]Department of Mathematics, Shah Abdul Latif University, Khairpur Mirs
Email: asimazad98@salu.edu.pk
[6]Assistant Professor Mathematics, Government Degree College, Pano Akil
Email: dostm641@gmail.com
[7]Lecturer Chemistry, Government Degree College, Pano Akil
Email: m.bachalkorai@gmail.com
[8]Lecturer Zoology, Government Degree College, Pano Akil
Email: bakhtiar.m.arab@gmail.com

DOI: https://doi.org/10.63163/jpehss.v4i1.1402

Abstract

The present study of T-type channel inserted with circular obstacle with no-slip boundary conditions. The circular obstacle is fitted on the center, its role is to divert fluid towards two outlets calculating the impact of circular obstacle on fluid, and also applied her finite element model(FEM), with basic governing equations known as Navier-Stoke equation and continuity equation in COMSOL MultiPhysics with the range of Reynolds numbers Re=01 to Re=2000. Analyze the effect of circular obstacle increasing Reynolds number on magnitude of velocity, streamline pattern, formation of vortex and impact of pressure on Junction type channel. Numerical analysis of T-type junction with circular obstacle and outlets of the channel and effect of surrounding region of the obstacle, and effect of pressure on two outlet and different wall of the T-type junction, the present investigation is very important in Biomedical and Engineering filed, such as mixing of fluids, cooling of different type of fluid, blood flow in arties, heat flow with obstacle.

Keywords: Finite Element Model, CFD, T-type Channel, Reynolds Number, No-slip Boundary Condition, Laminar Flow Pattern, Velocity Magnitude.

Introduction

Computational fluid dynamic (CFD) playing a crucial role in solving different computational problems related to biomedical and engineering fields using different geometries. The T-type channel is diverting fluid to two outlets. Numerical analysis of this data gives us accurate results. This principal is used in cooling systems, heat exchanger, fluid mixing and micro fluid system **[1].** The circular obstacle has such many roles; like as, vortex formation, pressure variation at outlets and change in streamline pattern in T-type channel **[2].** Most of the researcher are working on laminar and turbulent flow pattern with different obstacles and analyze it numerically. Dagtekin and Oztop **[3]** examined on the impact of fluid flow by the obstacle in the channel and the effect of recirculation of fluid and velocity distribution through the domain. Rahman et al.**[4]**. Impact of a channel with baffle-body blockage was fitted, further investigates numerically, by increasing and decreasing Reynolds Number effect on vortex near obstacle. Saha et al.**[5]**. Analyzed that the internal obstacle's plays role in flow divert and change in streamline patterns. Rashad **[6]**. Examine the effect of obstacle in T-type channel with Non-Newtonian fluid flow, different Reynolds numbers, under the various boundary conditions. Khanafer et al.**[7]**. Examine a channel with an obstacle, which has impacts on solid wall by the heigher Reynolds numbers with no-slip boundary conditions. The no-slip boundary conditions used in viscous fluid mechanics; by this the fluid velocity on the stationary surface wall becomes zero with respect to the wall. The wall shear behavior, viscous effect and boundary wall development in confined channel is predicted by this condition **[8]**. Numerically examine the strong velocity gradient and regions with circulation among the surroundings walls of obstacles by involving no-slip boundary conditions **[9]**. Governing equations are used in finite element model (FEM) for solving complicated problems related flow mechanics because of their accuracy and flexibility in using different geometries. CFD package helps us to give accurate results of velocity contours, pressure fields, streamline patterns and the vortex structures. Its important role is to solve problems numerically **[10]**. Examine, most researcher are using COMSOL MultiPhysics for the analysis of the effect of obstacle on fluid flow. This shows that the obstacle in a channel is effecting on the flow; in conclusion forming different streamline patterns **[11]**. Investigated the Range of Reynolds number is higher, then the size of vortex is increased and recirculation zone are observed by the help of inertial forces and flow instability effects.

In the present study two dimensional T-type channels, which is fixed with circular obstacle at the center of the channel is used in COMSOL MultiPhysics to examine incompressible laminar flow under no-slip boundary conditions. The main object of this study is to examine the effect of Reynolds number Re=01 to Re=2000, on the circular obstacle vortex formation, streamline patterns, pressure, and velocity magnitude inside the region of T-type channel. The results that will be obtained in T-type channel could help to design and optimizations of engineering and biomedical field such as, blood transportation, fluid transport, cooling systems, heat exchange, magnitude devices and further industrial and mechanical applications.

Problem Statement

The analysis of a T-type channel fixed with a circular obstacle which has much importance because of various applications such as heat exchanger, microfluidic devices, cooling phenomena and medical devices. It impacts on vortex pattern, streamline patterns, pressure contour and circular intensity in the inside of the T-type computational domain. The circular obstacle in T-type channel is diverting fluid towards two outlets and its behavior is important in the biomedical applications such as blood transportation through blocked arties by the obstructions. Furthermore, in recent study we are using incompressible laminar fluid flow in a T-type channel with a circular obstacle inserted at the center with no- slip boundary conditions applied in COMSOL MultiPhysics.

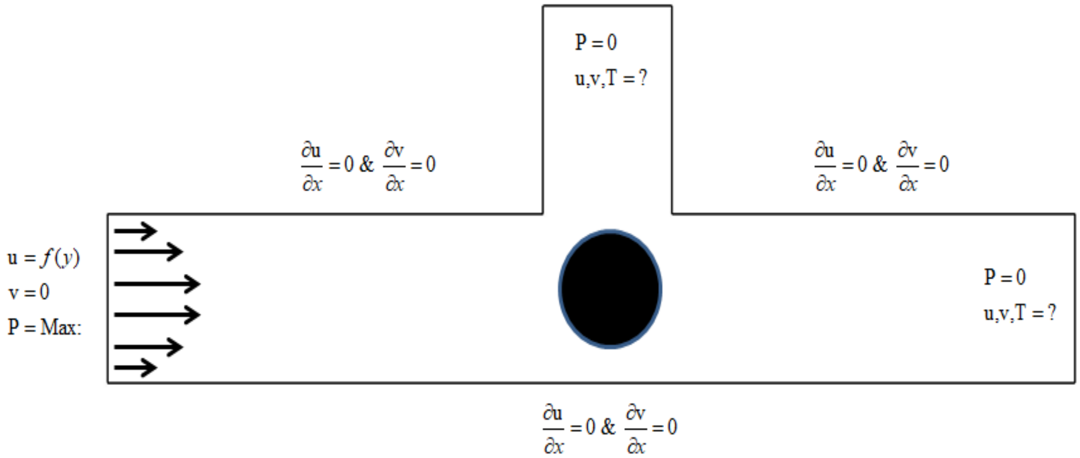

Fig:-01 Schematics domain of T-type channel fixed with circular obstacle

Fluid is allowed to pass through a T-type channel and which diverts toward outlets with effect on velocity magnitude, circular intensity, streamline pattern, pressure and vortex formation. The channels walls and circular obstacle satisfy the no-slip boundary conditions.

Governing Equations

Governing equations are constructed for incompressible laminar and turbulent fluid flow which is derived from the conservation law of fluid mechanics. The continuity and Navier-stokes equations are derived by the fundamental law of conservation fluid mechanics.

Continuity equation

This equation describes that mass of fluid in inter and leaving quantity of mass is equal. Continuity equation is for describing incompressible laminar fluid flow; which is given below,

$$\frac{\partial \mathrm{u}_1}{\partial x} + \frac{\partial \mathrm{v}_1}{\partial y} = 0$$

Momentum Equation

This equation is derived from the Newton's second law of motion. The rate of change of momentum of a fluid is equal to sum of forces that are acting on that fluid. Navier-Stokes equation is used for incompressible laminar fluid flow. It is given below,

X and Y directions

$$\frac{\partial u_1}{\partial t} = \frac{1}{\mathrm{Re}}\left(\frac{\partial^2 u_1}{\partial {x_1}^2} + \frac{\partial^2 u_1}{\partial {y_1}^2}\right) - u_1\left(\frac{\partial u_1}{\partial x_1} + \frac{\partial u_1}{\partial y_1}\right) - \frac{\partial P}{\partial x_1}$$

$$\frac{\partial u_2}{\partial t} = \frac{1}{\mathrm{Re}}\left(\frac{\partial^2 u_2}{\partial {x_1}^2} + \frac{\partial^2 u_2}{\partial {y_1}^2}\right) - u_2\left(\frac{\partial u_2}{\partial x_1} + \frac{\partial u_2}{\partial y_1}\right) - \frac{\partial P}{\partial y_1}$$

The forces acting equally in fluid are; on right side viscous diffusion forces and presser force; on left side of the equation are inertial forces and convective effects.

Reynolds Number

This is the dimensional less number and very important for flow behavior. It defines the ratio of internal and viscous forces.

$$\mathrm{Re} = \frac{\rho U L_{(total)}}{\mu}$$

The Reynolds number is changing Re=01 to Re=2000, to show the results of change in parameters that are streamline patterns, vortex formation, pressure, velocity magnitude and rotational flow rate.

Boundary Condition

Boundary conditions are applied in a domain with circular obstacle fixed. Inlet velocity profile is applied at inlet; whereas, pressure is zero at both primary outlet-I and secondary outlet-II outlets with circular obstacle.

$$u_1 = U\ ,\ v_1 = 0 \quad \& \quad P=0$$

The no-slip boundary conditions are exerted on the stationary walls and circular obstacle surface fixed at the center of the T-type channel.

$$u_1 = 0 \text{ and } v_1 = 0$$

All conditions help out as a tool for examining pressure variation, vortex formation and velocity distribution inside the T-type channel.

Numerical Method

For the computational analysis, Finite element model with varied Reynolds number are used in COSOL MultiPhysics. In a T-shaped channel a circular obstacle is fixed and laminar flow is crossed under no-slip boundary conditions. It is investigated that fluid diverts through obstacle.

Mesh Generation

Numerical analysis of non-uniform triangular finite helps to discretize the T-type domain, fluid diverts by the circular obstacle in the domain. For making the better results, extremely fine mesh used in COMSOL MultiPhysics. This method can give accurate velocity magnitude, vortex structure, pressure and circular zone are obtained, different meshes such as normal, fine, extra fine and extremely fine mashes are used for better results, and for decreasing the computational cost. Reynolds number are taken Re=01 to Re=2000, this helps to make convergence and numerical results better and accurate.

Mesh Independence Test

Mesh independence investigates better numerical results and convergence with different size of mesh. Quantity of mesh elements are increased near the obstacle and junction. By this analysis results are improved.

Results and Discussion

Streamline Distribution

Streamline contours effect express the difference in domain of T-type channel fixed with circular obstacle at the center. This diverts fluid towards two outlets. This analysis shows that with Re=100, there is no effect on streamline; whereas, after increasing to Re=500shows effect on streamlines, this form vortex near the walls of obstacle. More effect is appeared after increase to Re=1000 such as, circulation and fluid acceleration. Then increase to Re=1500 shows strong vortex structure and that represents intensity.

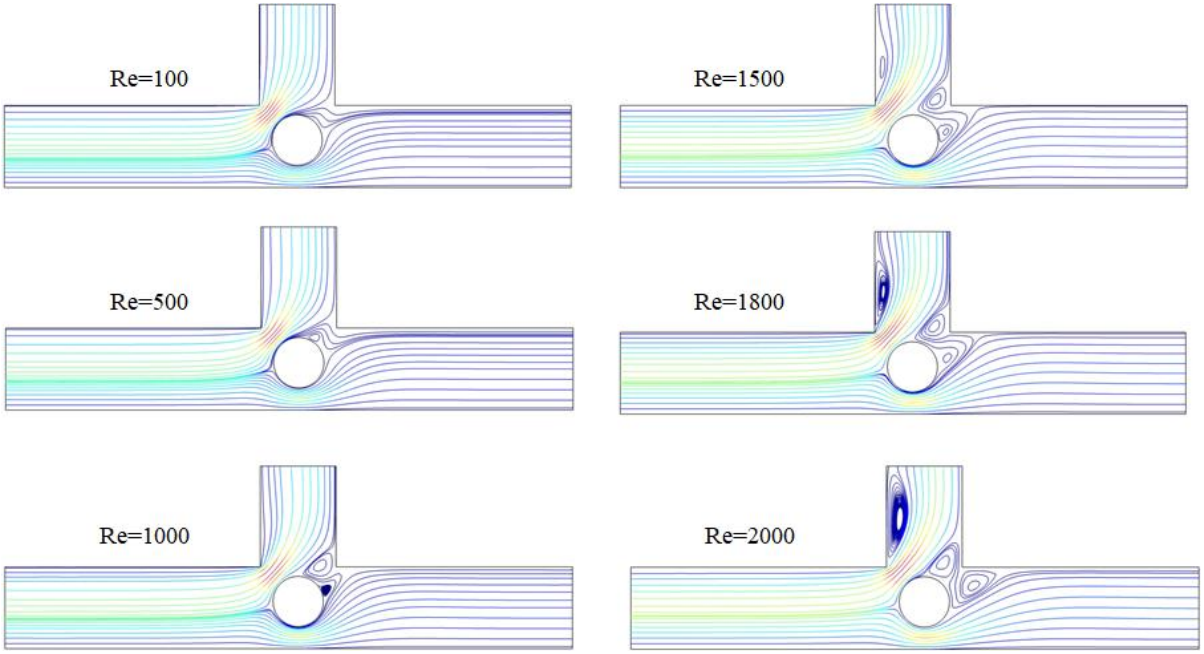

Fig:-02 Streamline patterns of various Reynolds numbers fixed with circular obstacle

Furthermore, increase to Re=2000 which increase size of vortices. This shows importance of circular obstacle in diverting of fluid towards two outlets. This computational analysis investigates the flow control and mixing enhancement. Impact of recirculation intensity is important in applications of Biomedical and Engineering fields.

Velocity Magnitude

Velocity magnitude profiles effect is expressed at outlets, primary outlet and secondary outlet with value of Re=100 is taken. Increase in Reynolds number to Re=2000, effects the flow behavior and this effect is consistent increase of Reynolds number to Re=2000, increase the velocity magnitude from 0.8×10^{-6} to $20.\times10^{-6}$ value. This effects 225% increased at the primary outlet-I. In addition velocity magnitude is 2.0×10^{-6} to $50.\times10^{-6}$ at the secondary outlet-II is upward in direction.

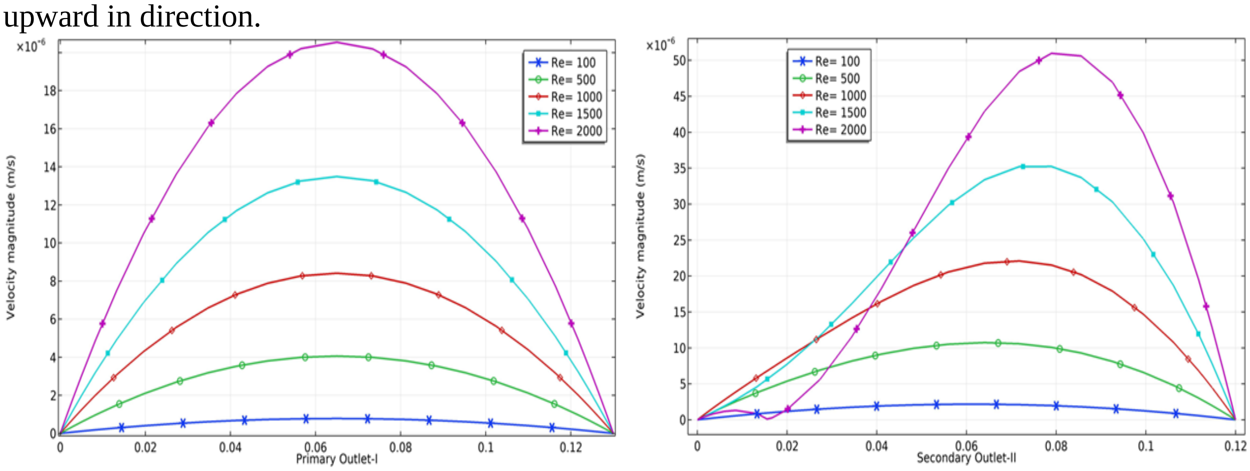

Fig- 03 Velocity magnitude results at various Reynolds numbers

It is showing that circular obstacle changes direction of flow more to the secondary outlet and also increase circular intensity. In this activity an internal effect, hydrodynamic and flow structure increase in T-type channel along with increase in Reynolds number.

Pressure distribution and pressure contours

T-type channel with a circular obstacle shown pressure contour and pressure gradients effects by increasing Reynolds number. Reynolds numbers increase in pressure variation; additionally, low pressure and higher pressure zone are created in T-type channel because of obstacle.

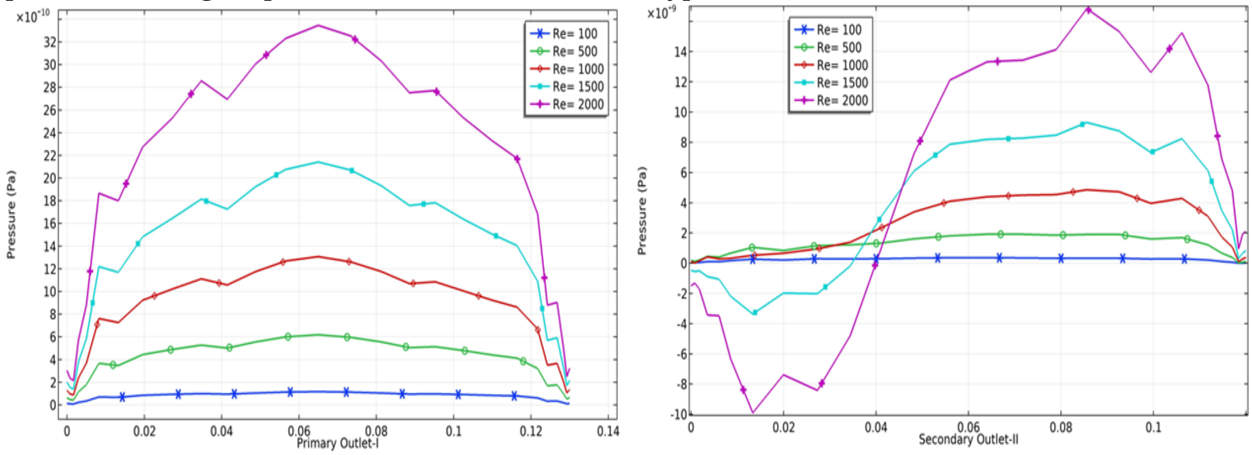

Fig:-04 Pressure with different Reynolds numbers at Primary and Secondary outlets

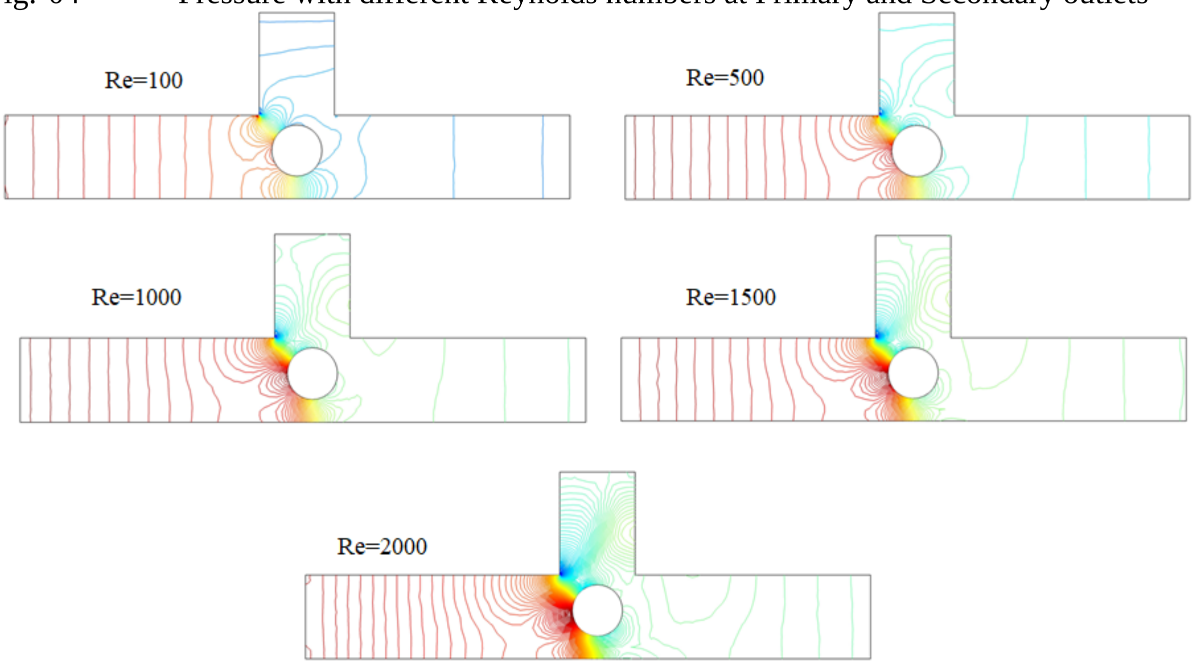

Fig:-05 Pressure contours with different Reynolds numbers Re=100 to Re=2000

Change in Reynolds number diverts fluid towards two outlets. In flow control, both the pressure distribution graph and pressure contours are altering better results. For control of fluid, results obtained by flow separation, vortex formation and Hydrodynamic performance are beneficial.

Rotational flow characteristics

It is provide that increase in Reynolds number rises circular intensity and rotational flow rate, by numerical results. The value of Rotational flow rate is 5.62×10^{-6} to 9.90×10^{-5}, vortex formation and divert increase to a great extent by increase to Re=2000.

Rotational flow rate		
Re	**Rotational flow rate**	
	($\Psi_{min:}$)	($\Psi_{max:}$)
100	1.54e-9	5.62e-6
200	2.83e-9	1.12e-5
400	5.53e-8	2.23e-5
600	1.80e-8	3.30e-5
800	6.48e-10	4.33e-5
1000	1.95e-11	5.31e-5
1200	2.35e-8	6.24e-5
1400	5.04e-9	7.14e-5
1800	6.35e-9	8.85e-5
2000	1.26e-8	9.9e-5

Table:-01 Velocity magnitude flow rate of T-type channel fixed with circular obstacle

Validation of Numerical Results

For checking accuracy of Numerical results, a validation process which aims to check the accuracy, reliability, validation and meshes with a numerical solution is held and analyzed by Finite element model(FEM) and COMSOL MultiPhysics. Different meshes such as, normal, fine, finer, extra fine and extremely fine are used and there results are analyzed by independence analysis. Results obtained by investigation are better results.

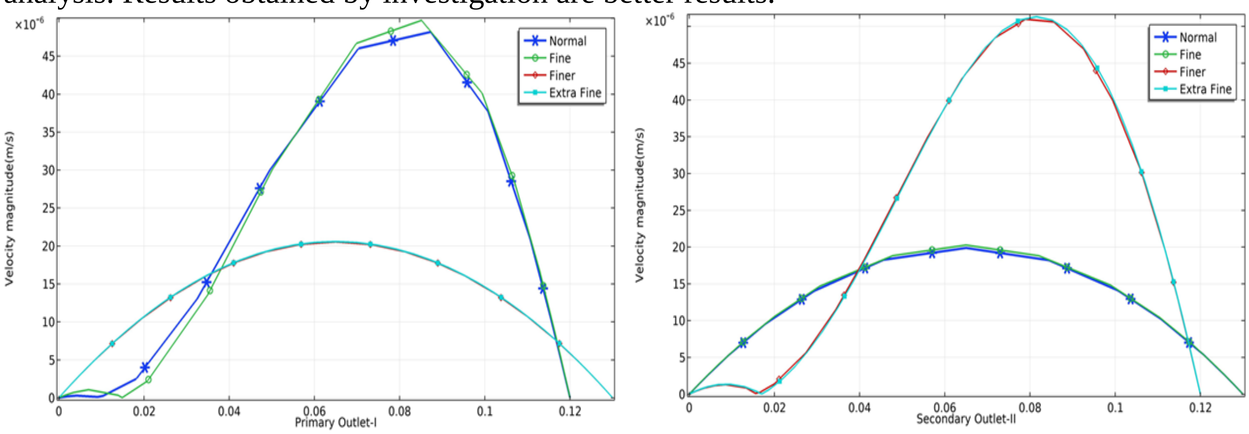

Fig:-06 Mesh independence studies at primary and secondary outlet of validation with laminar flow

T-type channel fixed with a circular obstacle and computational model of incompressible laminar flow gives validation that results. T-type channel fixed with a circular obstacle and computational model of incompressible laminar flow gives validation that results obtained by investigation shows that results are beneficial and consistent.

Conclusion

The present study is having no-slip boundary condition in T-type channel and with circular obstacle fixed at the center. Incompressible laminar floe with help of COMSOL MultiPhysics with Finite element Method and using range of Reynolds number from Re=01 to Re=2000 value. It is expressed by the results that fluid diverts towards two outlets by the circular obstacle and this is because of hydrodynamic behavior. Furthermore, increase in Reynolds number causes change in vortex, circular intensity, fluid acceleration and pressure contours. Validation analysis and mesh independence results are obtained by finite element model (FEM) with help of COMSOL MultiPhysics well better and accurate. This investigation provides information for the fluid transport phenomena and blood circulation in arteries. In conclusion, it can play a major role in cooling system, heat exchange, microfluidic devices, biomedical flow application and engineering fields.

References

T. Iwatsu, J. M. Hyun, and K. Kuwahara, "Mixed convection in a driven cavity with a stable vertical temperature gradient," *International Journal of Heat and Mass Transfer*, vol. 36, no. 6, pp. 1601–1608, 1993.

Y. Varol, H. F. Oztop, and A. Koca, "Entropy generation due to conjugate natural convection in enclosures with partial heating," *International Communications in Heat and Mass Transfer*, vol. 35, no. 4, pp. 492–498, 2008.

H. F. Oztop and I. Dagtekin, "Mixed convection in two-sided lid-driven differentially heated square cavity," *International Journal of Heat and Mass Transfer*, vol. 47, no. 8–9, pp. 1761–1769, 2004.

M. M. Rahman,M. A. Alim, and M. K. Chowdhury, "Numerical study of mixed convection in a ventilated cavity with a heat-generating obstacle," *Computers & Mathematics with Applications*, vol. 59, no. 2, pp. 804–816, 2010.

S. C. Saha, M. Mamun, and G. Guo, "Effect of internal obstacles on fluid flow and heat transfer in channels," *International Communications in Heat and Mass Transfer*, vol. 34, no. 5, pp. 605–615, 2007.

A. M. Rashad,"Influence of magnetic field and slip condition on non-Newtonian fluid flow," *Applied Mathematics and Computation*, vol. 219, no. 12, pp. 6844–6858, 2013.

K. Khanafer, K. Vafai, and M. Lightstone, "Buoyancy-driven heat transfer enhancement in a two-dimensional enclosure utilizing nanofluids," *International Journal of Heat and Mass Transfer*, vol. 46, no. 19, pp. 3639–3653, 2003.

M. Sheikholeslami, M. Gorji-Bandpy, and D. D. Ganji, "Numerical investigation of MHD effects on nanofluid heat transfer," *Thermal Science*, vol. 17, no. 4, pp. 1151–1163, 2013.

N. Aminossadati and B. Ghasemi, "Natural convection cooling of a localized heat source at the bottom of a nanofluid-filled enclosure, "*European Journal of Mechanics B/Fluids*, vol. 28, no. 5, pp. 630–640, 2009.

J. Donea and A. Huerta,"Finite element methods for flow problems," *Computers & Fluids*, vol. 30, no. 1, pp. 1–30, 2001.

M. Rahimi and M. Abbassi, "Numerical simulation of laminar flow in channels with obstacles using finite element methods," *Applied Mathematical Modelling*, vol. 35, no. 9, pp. 4268–4280, 2011.

S. Parvin, R. Nasrin, N. Alim, and M. A. Hossain, "Effect of Reynolds number on flow characteristics in a channel with internal obstacles," *International Journal of Mechanical and Materials Engineering*, vol. 7, no. 2, pp. 151–159, 2012.